Gunthard Kraus, DG8GB

Modern Design of Stripline Low Pass Filters

The subject of low pass filters is covered through a comparison of designs, with the assistance of contemporary development tools based on PC programs. The special features of Micro Stripline filters are also referred to.

1. Introduction

The current situation and recent progress in relation to the design of stripline band pass filters have already been discussed here The present article is devoted to low pass filters. In the first section, the work is done using the tried and tested microwave CAD program PUFF, while in the second section the design work is repeated using the most recent student version of ANSOFT Serenade. The results can thus be compared with one another and the progress made in microwave CADs and in circuit simulation can be evaluated Finally, this subject is rounded off with measurements on the assembled specimen circuits.

2. Filter design using PUFF

2.1. Design principles for stripline filters

The filter type chosen is the Chebyschev low pass (for more detailed information on the selection of filters of a suitable type and grade, see the article referred to in [1]).

The following values are given:

Filter grade n	**Number of poles = number of components = 5**
π format	**(Shut capacitors with series inductors)**
Impedance Z	**50Ω**
Maximum ripple of S21 in transmission range	**0.1dB**
Cut off frequency	**1,700 MHz**

The components required are calculated in accordance with these parameters, using the tried and tested filter program “fds.zip” (Fig. 1). We therefore need two capacitors, each with 2.147pF, a capacitor with 3.67pF and two coils, each with 6.42nH.

Now we use the well known method for implementation (Fig. 2):

The capacitors are created using short but

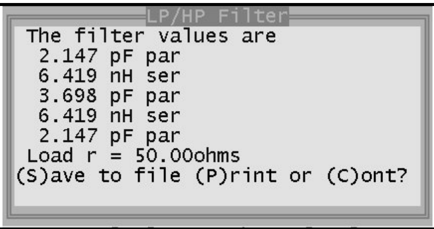

Fig 1: Filter design using fds.zip for cut off frequency of 1700MHz.

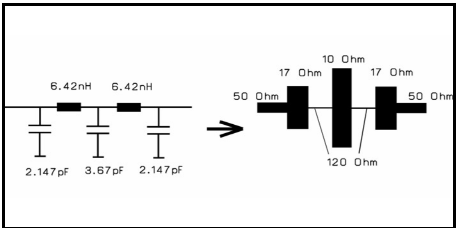

Fig 2: Schematic of low low pass filter.

very wide stripline sections with an impedance of less than 20Ω. This ensures that the inductive fraction of the line plays no great role. Very thin, short lines (with Z = at least 100Ω) are used to produce the inductances, provided the capacitive fractions are still sufficiently low.

For successful design, stick to the following rules, which originate from practical experience in filter construction:

The electrical lengths of the line sections should lie between 10 and 30 degrees when the cut off frequency of the filter is used as the design frequency of the line.

For the "capacitors", the impedance level of the line sections used for a 50Ω system should not exceed 20Ω. For the "coils", select lines with at least Z = 100Ω.

For the "wide capacitor lines", the width to length ratio should not exceed 8.

For the "thin inductance lines", stick as close as possible to a minimum conductor width of approximately 0.2 mm as the lower limit. This results in a high inductance and thus short line lengths. Unfortunately such extremely thin tracks are notable for their high losses; the lines therefore have to be made slightly wider, which makes it easier to manufacture the printed circuit board.

But we must be clear about one thing: Replacing genuine capacitors and coils with line sections does not immediately lead to a perfect solution, since it is well known that any line consists of inductive and capacitive parts. In addition, transformed lines also have resistances, the source resistance and the load resistance. Consequently discrepancies are possible on the first simulation run, thus optimisation is indispensable.

2.2. Determination of line data

Use a text editor to open the SETUP file for PUFF, in order to enter and then store the design frequency, fd, together with the data for the board material used (Rogers R04003, with a thickness of 32 MIL). They are shown in bold in the following file extract:

\b{oard} .puf file for PUFF, version 2.1d}
d 0 {display: 0 VGA or PUFF chooses, 1 EGA}
o 1 {artwork output format: 0 dot-matrix, 1 Laser Jet, 2 HPGL file}
t 0 {type: 0 for microstrip, 1 for stripline, 2 for Manhattan}
zd 50.000 Ohms {normalizing impedance. 0<zd}
fd 1.7 GHz {design frequency. 0<fd}
er 3.380 {dielectric constant. er>0}
h 0.813 mm {dielectric thickness. h>0}
s 100.000 mm {circuit-board side length. s>0}
c 100.000 mm {connector separation. c>=0}
r 0.010 mm {circuit resolution, r>0, use Um for micrometers}
a 0.000 mm {artwork width correction.}
mt 35 Um {metal thickness, use Um for micrometers.}
sr 5.000 Um {metal surface roughness, use Um for micrometers.}
lt 1.0E-0003 {dielectric loss tangent.}
cd 5.8E+0007 {conductivity of metal in mhos/meter.}
p 2.000 {photographic reduction ratio. p<=203.2mm/s}
m 0.600 {mitering fraction. 0<=m<1}

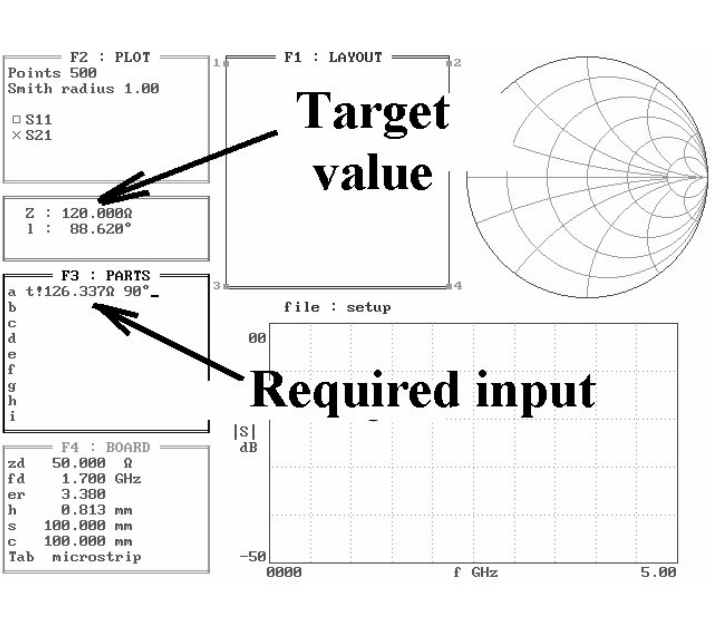

Fig 3: Adjusting the input to obtain the required impedance of 120Ω.

Next use a width of 0.25 mm for the “inductance line section”, thus there is no great problems in producing the printed circuit board.

Then PUFF is started to determine the impedance levels for the conductor widths. This is done in the following manner. A real model transmission line is entered in field F3, with an impedance level of, e.g., 100Ω and an electrical length of 90 degrees. The “real modelling mode”, as is well known, is invoked using the exclamation mark after the letter “t”. If the equals sign is now pressed, then the actual impedance level and the actual electrical length appear in the dialogue field. The entry value is altered until an actual impedance level of 120Ω appears in the dialogue field. As Fig. 3 shows, it is necessary to increase the input value to 126.337Ω? to achieve this. If we now delete the exclamation mark and press the equals sign again, we see the mechanical data for this line. Fig. 4 shows that the width of 0.25 mm is well attained.

In order to add the two outer 2.147pF capacitors, we use lines with an impedance of Z = 17Ω, whilst for the central capacitor (3.67pF) a wider line with Z = 10Ω makes better sense. If the line is too long, this unfortunately makes the attenuation worse at very high frequencies, which also accounts for the recommendation for an electrical line length of 10 to 30 degrees. We use PUFF for these two lines as well, to determine the necessary conductor widths. We should also include the 50Ω feed lines to connect the filter structure with the connection sockets.

If we keep strictly to the methods referred to, we finally obtain the entry values in accordance with Fig. 5. If we finally remove the exclamation mark in each line and enter the equals sign, we can compile the first little table:

Line	**50 Ω**	**10 Ω**	**17 Ω**	**120 Ω**
Conductor width (mm)	**1.84**	**14.62**	**7.94**	**0.25**

We now continue using simple approximation formulae to determine the required line lengths. Here we start with the narrow lines serving as coils (w =

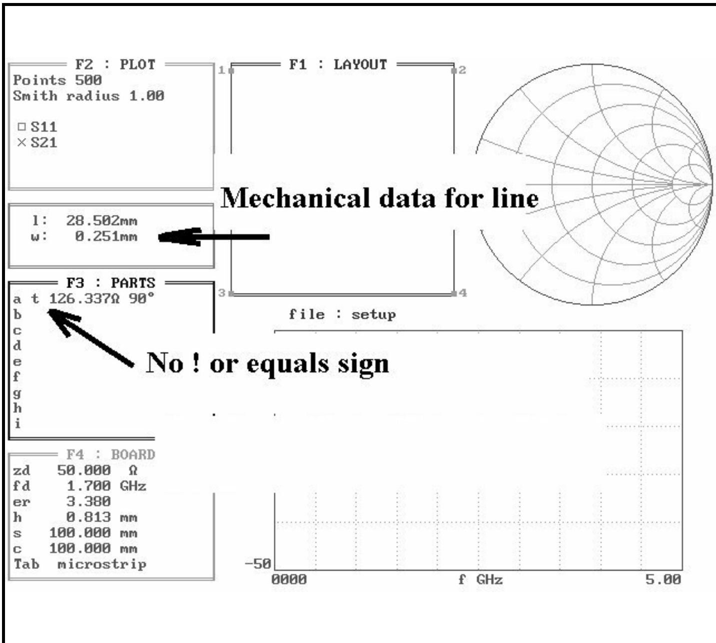

Fig 4: Remove the equals sign to calculate the mechanical data for the line.

0.25 mm), where:

$$\frac{l_{L-Line}}{\lambda} = \frac{f \cdot L}{Z_L}$$

ZL is the selected impedance for the “inductive line”, f is the actual frequency and L is the required inductance.

For f = 1700MHz, L = 6.419nH and ZL= 120Ω we obtain the following standardised length:

$$\frac{L_{L-Line}}{\lambda} = \frac{1700 \cdot 6.41}{120} = 0.090935$$

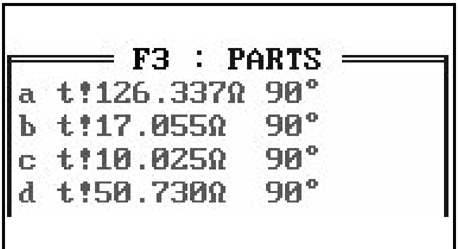

Fig 5: Real inputs for 120Ω, 17Ω, 10Ω and 50Ω lines.

Since a complete wavelength corresponds to an angle of 360 degrees, this line section will have an electrical length of 0.090935 x 360 degrees = 32.74 degrees. We obtain the associated mechanical length through the well known procedure using PUFF:

Put an exclamation mark behind “t”, enter the length, and continue to correct until this value is finally obtained when the equals sign is entered (Fig. 6). Now delete the exclamation mark and press the equals sign again to obtain the figures for the mechanical dimensions. This gives a physical length of 10.53 mm.

Summary of complete line data for inductances with a design frequency of 1,700 MHz:

Z	**120 Ω**
Electrical length	**32,74 Degree**
Mechanical length	**10.53 mm**
Conductor width	**0.25 mm**

The capacitor line calculation requires the following approximation formula:

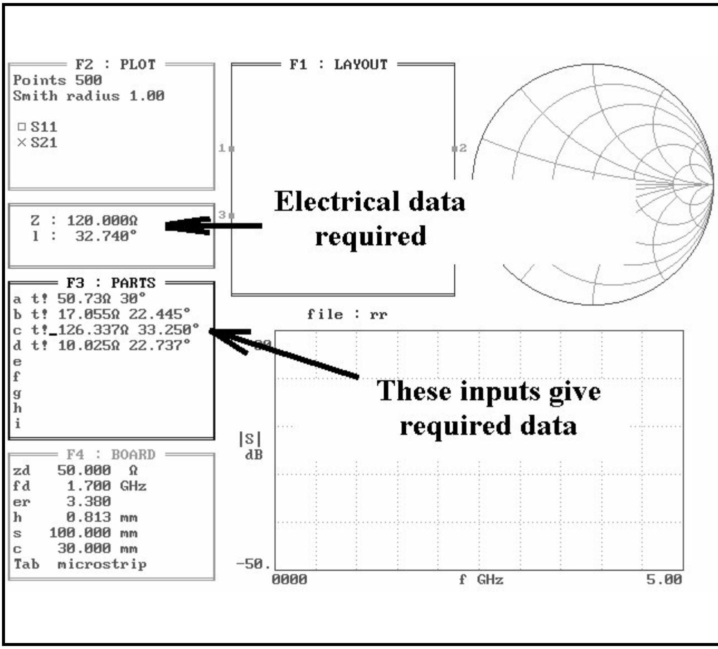

Fig 6: Calculation of the electrical length of the lines.

$$\frac{I_{C-Line}}{\lambda} = f \cdot C \cdot Z_C$$

where Zc is the impedance of the capacitor line.

For the first and third capacitors with C = 2.147pF, at 1,700MHz we obtain:

$$\frac{I_{C-Line}}{\lambda} = 1700 \cdot 2.147 \cdot 17 = 0.062$$

That gives an electrical length for the line of 0.062 x 360 degrees = 22.32 degrees and in addition PUFF supplies a conductor length of 6.31 mm with a conductor width of 7.94 mm.

Summary of full line data for first and third capacities for a design frequency of 1,700MHz:

Z	**17 Ω**
Electrical length	**22.32 degrees**
Mechanical length	**6.31 mm**
Conductor width	**7.94 mm**

It is frequently necessary to repeat this run for the central capacitor of 3.698 pF:

$$\frac{I_{C-Line}}{\lambda} = 1700 \cdot 3.698 \cdot 10 = 0.082866$$

That gives an electrical length for the line of 0.062866 x 360 degrees = 22.63 degrees and a conductor length of 6.29 mm with a conductor width of 14.62 mm.

Summary of full line data for central capacity for a design frequency of 1,700MHz:

Z	**10 Ω**
Electrical length	**22.63 degrees**
Physical length	**6.29 mm**
Conductor width	**14.62 mm**

2.3. Determination of substitute data for stripline steps

In the existing circuit, a repeated change (Impedance Step) is carried out between wide and narrow stripline sections. But each individual step acts as an irregularity, the approximate effect of these can be determined by using a substitute circuit made from a series inductance and a case capacitance (like a simple low pass filter!).

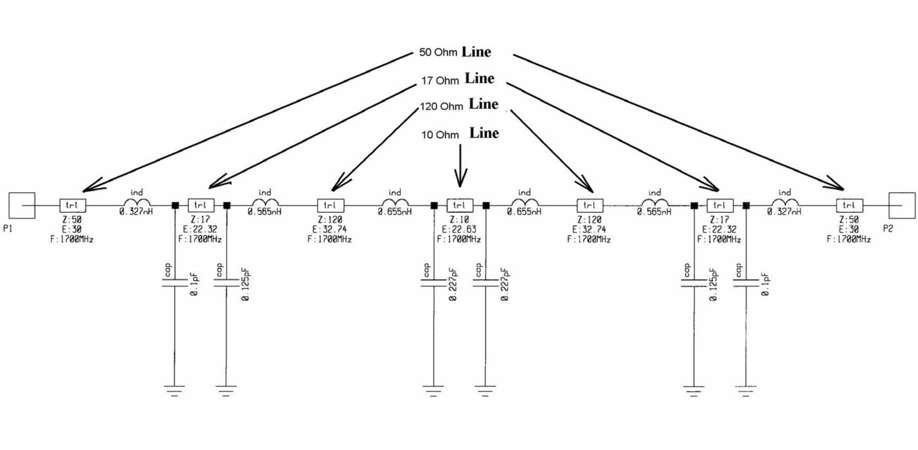

Fig 7: Circuit diagram of low pass filter produced using ANSOFT Seranade, showing impedance steps.

If the simulated circuit is to correspond precisely to the parameters (as far as possible) without a lot of reworking, and at the same time be capable of use, we can not avoid the need for these circuit extensions. But they are already in existence in a modern microwave CAD program as modules and the correction is thus relatively simple. PUFF is unfortunately not that well developed yet, so the calculation formulae from [4] are used.

In order to specify these substitute components, we require not only the impedance values and conductor widths of each line section, but also the associated effective dielectric constants. These constants are determined and used by PUFF during its calculations, but unfortunately they are not output. So here we have to outwit the program:

First step:

Use a pocket calculator to work out the free space wavelength for the frequency of 1,700MHz

$$\lambda = \frac{c}{f} = \frac{3.10^8}{1700} = 176.47mm$$

Second step:

Increase the inputs on all four lines in field F3, one after another, until the real lengths are precisely 360 degrees when the equals sign is pressed (i.e. one wavelength). Then delete the exclamation mark in each line and enter the equals sign instead. This will give you the corresponding physical lengths. They are:

Line	**50 Ω**	**10 Ω**	**17 Ω**	**120 Ω**
Conductor length	**108.496**	**99.998**	**102.004**	**115.785**

Third step:

Using the following simple rearranged short cut formula,

$$\varepsilon_{eff} = \left(\frac{\lambda_{air}}{\lambda_{Line}} \right)^2$$

All the effective dielectric constants can now be determined using the pocket calculator:

Line	**50 Ω**	**10 Ω**	**17 Ω**	**120 Ω**
Conductor width	**1.84**	**14.62**	**7.94**	**0.25**
εeff	**2,646**	**3,114**	**2,993**	**2 ,323**

All the step components can now be specified using this data. The series inductance at each step can be deter-

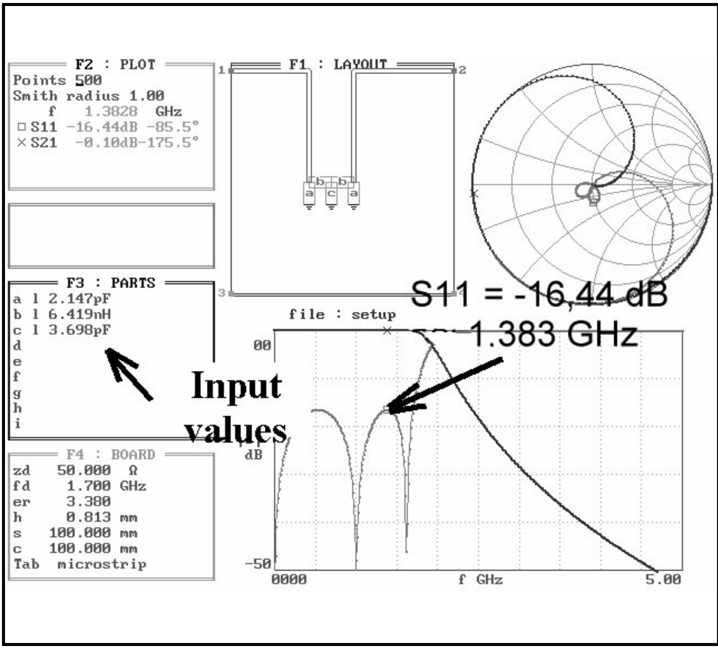

Fig 8: Results of simulating ideal filter.

mined using the following approximation formula:

$$L = 0.000987 \cdot h \cdot \left(1 - \frac{Z_1}{Z_2} \cdot \frac{\sqrt{\varepsilon_{1eff}}}{\sqrt{\varepsilon_{2eff}}} \right)$$

where: h is the board thickness in µm, Z1 the impedance level of the wide line, Z2 the impedance level of the narrow line, ε_{1eff} the effective dielectric constant of of the wide line on the board, ε_{2eff} the effective dielectric constant of the narrow line on the board. The inductance is in nH.

Thus we can once again compile a table:

Transition from	**50 Ω to 17 Ω**	**17 Ω to 120 Ω**	**120 Ω to 10 Ω**
Inductance	**1.84 nH**	**14.62 nH**	**7.94 nH**

It is somewhat more laborious to calculate the case capacitance arising:

$$C = 0.00137 \cdot \frac{\sqrt{\varepsilon_{1eff}}}{Z_1} \cdot \left(1 - \frac{W_2}{W_1} \right) \cdot h \cdot \left(\frac{\varepsilon_{1eff} + 0.3}{\varepsilon_{1eff} - 0.258} \right) \cdot \left(\frac{\frac{W_1}{h} + 0.264}{\frac{W_1}{h} + 0.8} \right)$$

where: Z1 = Impedance of the wide line, ε_{1eff} = effective dielectric constant of wide line, W1 = conductor width of wide line, W2 = conductor width of narrow line, h = board thickness in µm (here: 0.813 mm = 813 µm). The capacity is in pF.

With the above data, we obtain a second table using the pocket calculator:

Transition from	**50 Ω to 17 Ω**	**17 Ω to 120 Ω**	**120 Ω to 10 Ω**
Capacity	**0,1 pF**	**0,125 pF**	**0,227 pF**

When drawing the simulation wiring diagram in the next chapter, please bear in mind that:

series inductances are always connected directly to the narrower line section, whilst the shunt capacitances are always mounted parallel to the wider line section!

The circuit diagram that applies to the simulation, with all the irregularities is shown in Fig. 7. It was drawn using the circuit editor of ANSOFT Serenade and

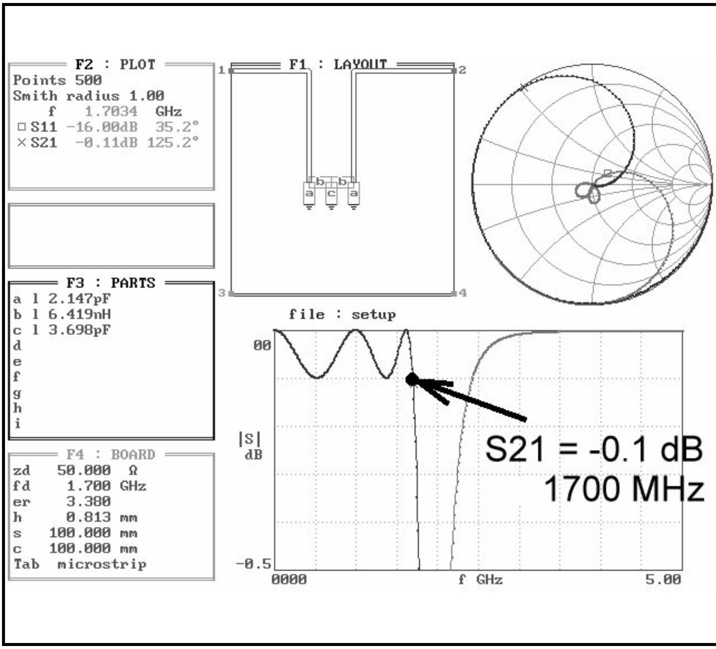

Fig 9: Magnified view of S21 curve.

immediately gives an impression of the layout of this modern program. But the number of components required is not very far off the maximum of 25 components that can be used with the free student version of ANSOFT Serenade.

2.4. Simulation and optimisation of circuit

First we have to simulate the ideal low pass filter made from coils and capacitors, in order to have the optimisation target visible. The result can be seen in Fig. 8, where special attention should be paid to the precise heights of the two “camels humps” of S11. S11 rises right up to -16.44 dB since it is associated with the S21 ripple of -0.1 dB). If this value is obtained again following the optimisation of the stripline circuit, the ripple is also tuned, being approximately -0.1 dB in the transmission range. It is increased merely by the attenuation, that rises with the frequency, due to the printed circuit board losses and the skin effect, etc.

The ideal curve for S21 is shown in Fig. 9 on a greatly expanded scale. From now on, these two sheets should be kept next to the PC for the work that follows.

We now enter all the components required for the circuit in field F3. Make sure you dont forget the exclamation mark for all line sections. See Fig. 6 for the precise data for the lines. Once everything has been entered, the circuit is correctly assembled in F1, then switched to F2 and simulated. The simulation result from S11 and S21 in the frequency range from 0 to 5GHz and the amplitude range from 0 to 1dB for monitoring the ripple is shown in Fig. 10. In Fig. 11 the amplitude range between 0 and -50dB has been selected in order to check the two S11 humps. It can be seen that the impedance leaps have the most important effect, and also that the cut off frequency has fallen by almost 400MHz.

But dont worry: optimisation is not especially difficult, for you merely have to balance out the influence of the additional inductances:

- The camel hump at the bottom right hand corner can be lifted by means

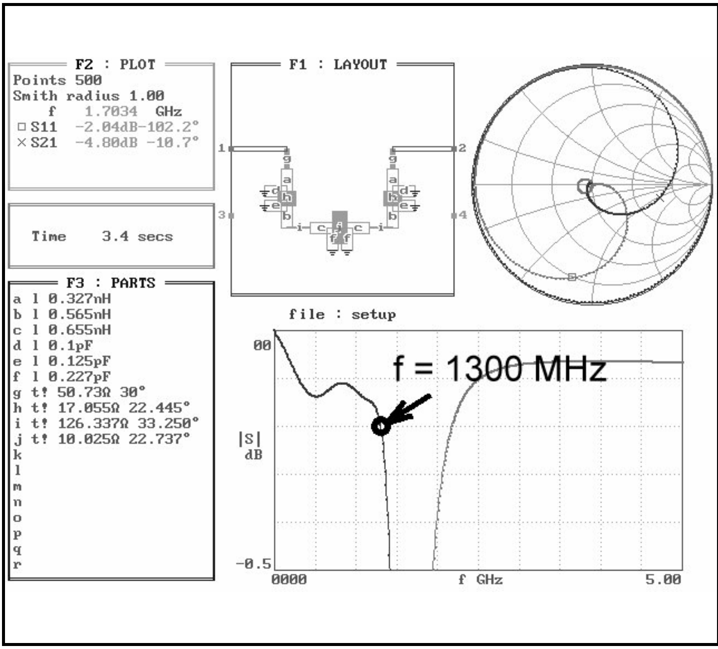

Fig 10: Simulation results from 0 to 5GHz.

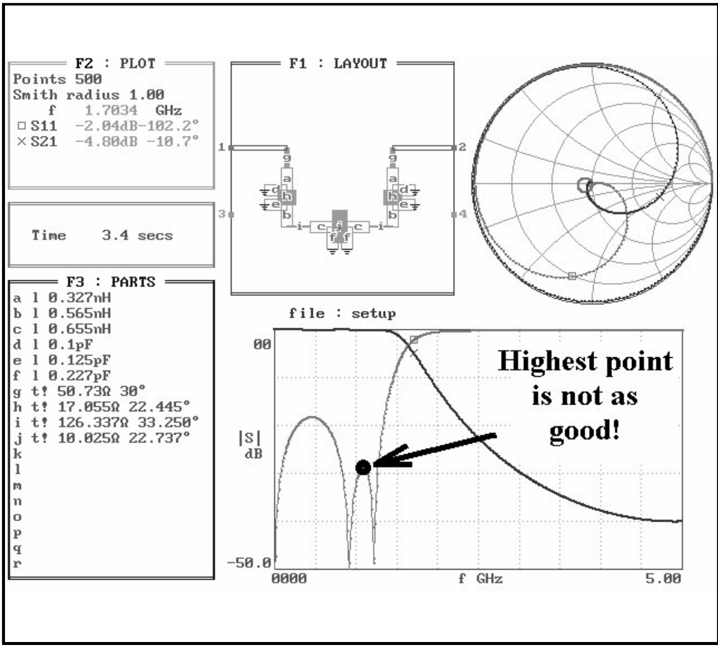

Fig 11: Expanded view of 0 to 5GHz results.

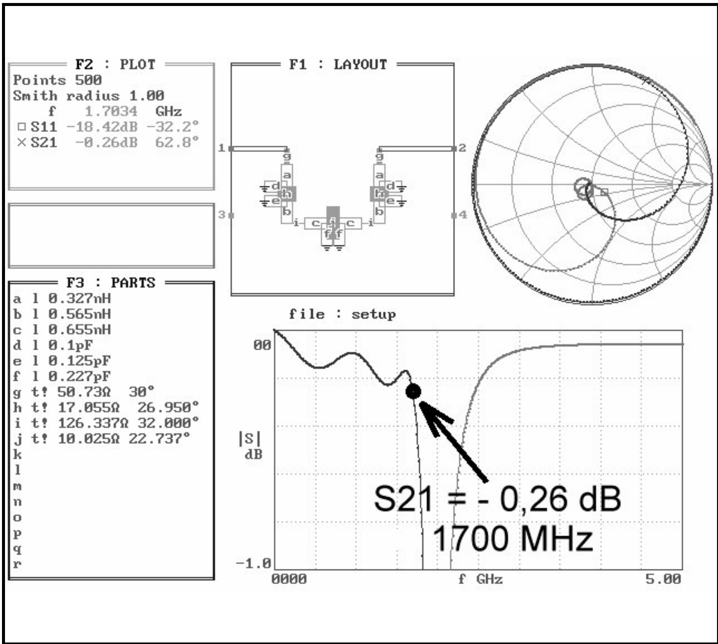

Fig 12: Results showing S21 curve after optimisation.

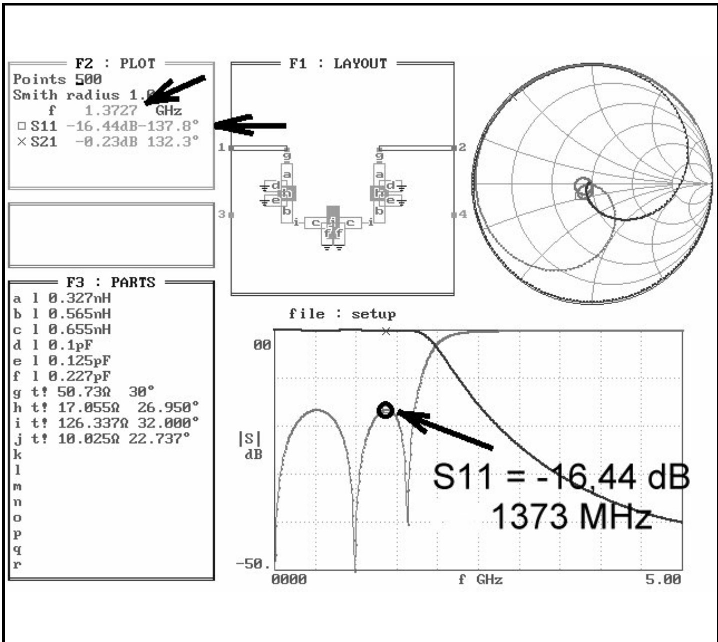

Fig 13: Results showing S11 curve after optimisation.

of an extension of the two external 17Ω line sections.

- Then, if necessary, the coils (the two thin 120Ω line sections) are slightly shortened, and in this way the two humps become the same height (target = -16.44dB).

Once this has been done, the display range for S21 and S11 is restricted to 0 to 1dB and simulated again. The pre set ripple value of 0.1dB can then be very easily recognised on the fundamental attenuation, which rises as the frequency increases. Yet the cut off frequency of the filter is still much too low. We therefore simply increase the design frequency in field F4 until, at 1,700MHz, we can observe the sharp transition from the transmission range into the filter attenuation band. This is at fd = 2,100MHz in this case. Once again, we convert this to the display range between 0 and 50dB using a linear plot diagram and the precise height of the two humps is checked with the help of the cursor. Any small discrepancies present are eliminated in accordance with methods a) and b). The results can be seen in the two diagrams, Figs. 12 and 13.

Finally, another table can be compiled, in which the definitive mechanical and electrical data for the individual line sections are entered after optimisation. These are not needed for the draft layout, but they can be used as a way in to monitor this PUFF design in the second part, using ANSOFT Serenade.

Make sure that:

- The design frequency has been increased to 2,100MHz and that the matter of the open end extension (the increase in the line length caused by projecting field lines) has also been dealt with automatically through the insertion of the impedance steps.
- The pre set mechanical lengths and widths for the line sections may therefore be transferred directly into the printed circuit board layout without any further correction! See Table 1 below.

2.5. PUFF versus HARMONICA

Using PUFF we can switch, by means of a simple exclamation mark, from ideal modelling (entering impedance level and electrical length at design frequency) to real modelling (with physical length and width, together with all the line losses).

ANSOFT unfortunately separates these two options completely and even uses different graphic symbols in the two cases. So separate wiring diagrams and projects must also be drawn up for it!

Here we can go straight to the simulation of physical dimensions in the first run. The original PUFF simulation is investigated (housing cover not taken into account). In the second run a space of 13mm between the board and the cover is used which simulates typical apparatus and its influence determined.

2.5.1 Simulation check of PUFF design with SERENADE

Serenade is started up, a new project is set up and then the circuit is drawn. Assistance should be obtained from Fig.

Table 1: Pre set mechanical dimensions.

	50Ω line	10Ω line	17Ω line	120Ω line
Width	1.84mm	14.62mm	7.94mm	0.25mm
Length		5.09mm	6.15mm	8.20mm
Electrical length in deg. at 2100MHz		22.63	26.8	31.51

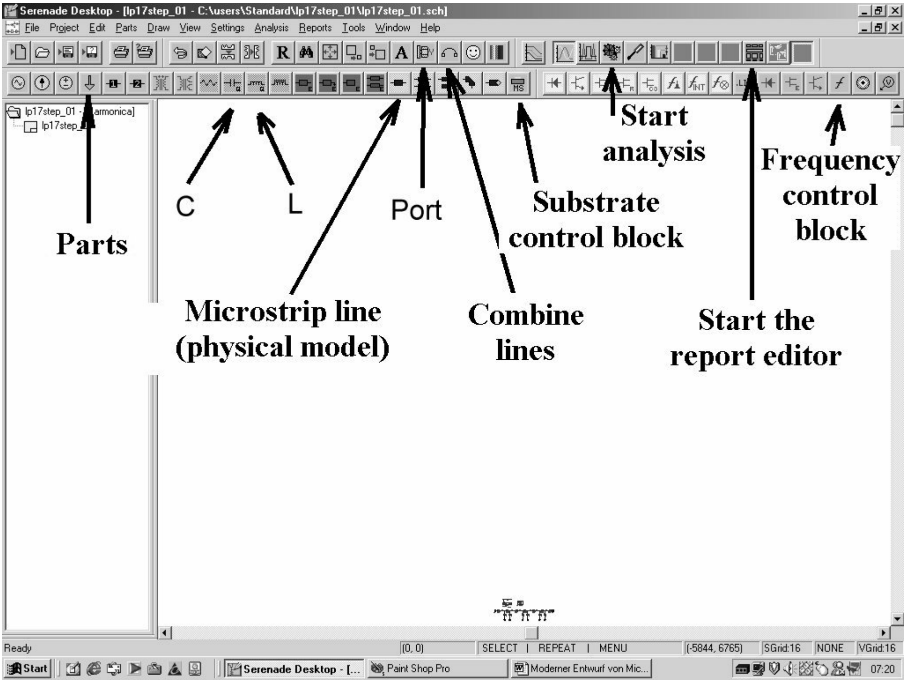

Fig 14: The opening window of Seranade showing main items of interest.

14 here. It shows the buttons behind which the various components and elements are hidden. The correct entries in the substrate screen can be seen in Fig. 15 (board and housing data), where the unit mm should not be forgotten, for all dimensions. Fig. 16 shows the circuit ready for simulation, including the frequency module for the sweep from 0 to 5GHz in 5MHz steps.

As soon as all this has been done, the analysis button (button with the little gear on top) can be pressed. At first nothing can be seen, until the turquoise / grey Report Editor button is pressed, this gives the option of outputting the results in diagram form. S11 and S21 are shown in dB as set up by the method shown in Fig. 17. Fig. 18 shows, on a greatly enlarged scale, the result of the simulation. Both the shape of the S21 curve and the ripple cut off frequency entered (with attention being paid to the fundamental filter attenuation!) tally completely with Fig. 12, the simulation using PUFF. This can also be seen in Fig. 19, here the results are represented without magnification and thus the shape of the S11 curve can be checked. A comparison with Fig. 13 gives a big boost to our trust in PUFF.

2.5.2. Simulation using housing data

To do this we first call upon the Stripline Calculator TRL85 and calculate the corrected physical line data for an interval of 13 mm between board and cover. The first step in doing this is naturally to prepare the line and board targets required:

First use the line data for the simulation using PUFF at a design frequency of 2,100MHz (see Table 1) and then the technical data for the printed circuit board. The interval referred to between

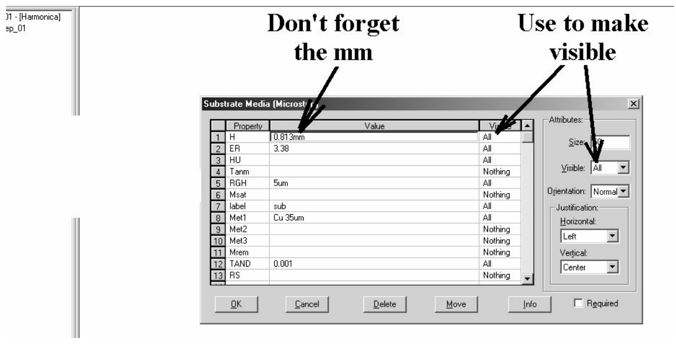

Fig 15: Entry of substrate data.

the cover and the board is added here as well (= marked in bold):

Material:	Rogers R04003
Board thickness:	H = 32 MIL = 0.813 mm
Dielectric constant:	er = 3.38
Loss factor:	TAND = 0.001
Copper coating:	TH = 35 µm
Surface roughness:	RGH = 5 µm
Distance between cover and board:	**HU = 13 mm**

The determining the new values for the first line section (Z = 17Ω, electrical length = 26.8 degrees at 2,100MHz) and, once again, the steps required to get there are shown in Fig. 20. If we repeat that for the other lines, then we finally obtain the results table (Table 2).

Anyone who compares the two tables (with and without housing) is immediately struck by the fact that including the cover distance referred to affects only the wide line sections, their conductor width is reduced. This is easy to understand, since some additional electrical field lines are pulled up by these large plates. This increases the capacity and must be corrected, whereas for the short "fips", showing the widths of 120Ω and 0.25 mm, scarcely anything is noticeable.

Now the previous HARMONICA project is re started and every physical line value in it is checked and / or corrected in

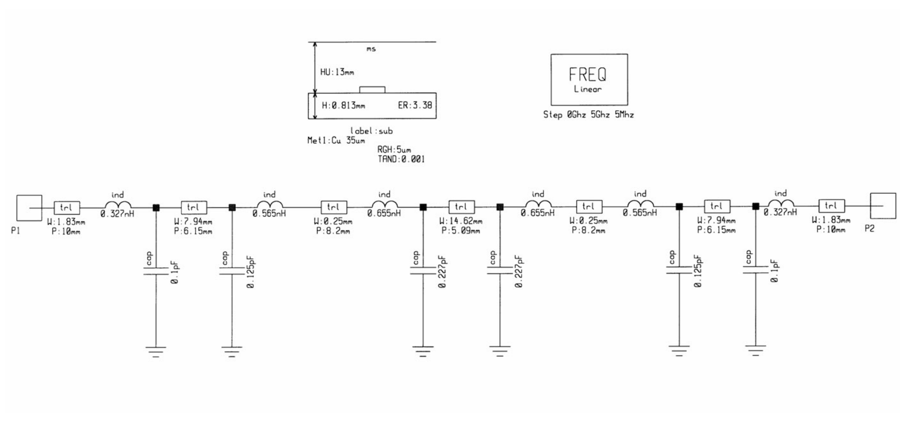

Fig 16: Circuit diagram ready for simulation.

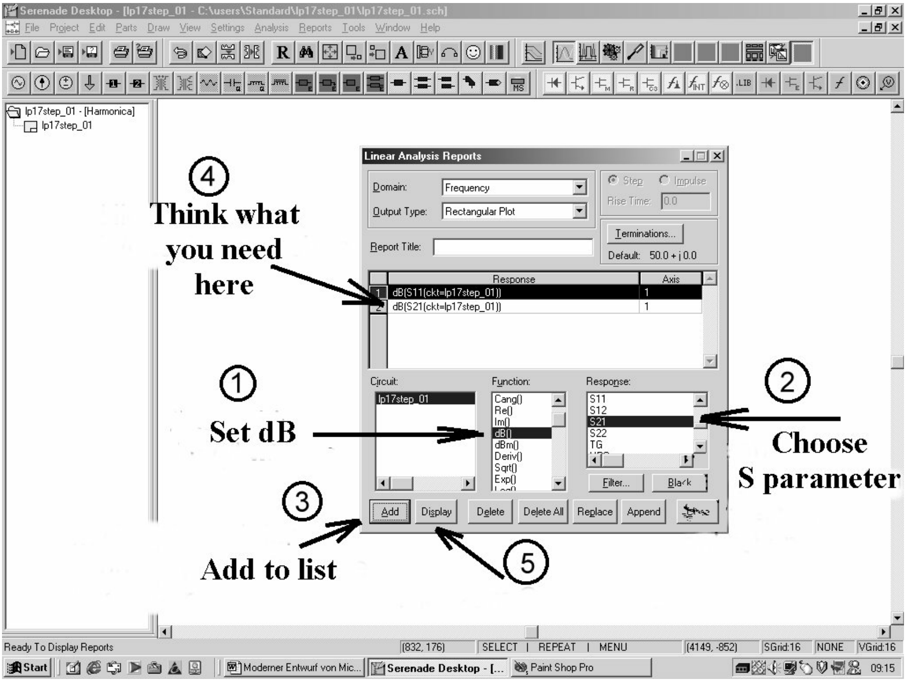

Fig 17: Setting up reports editor.

accordance with the table.

The new entry "HU = 13 mm" for the substrate control block is very important here, and should not be forgotten!

If we now start the simulation and extract and magnify S21 (Fig. 21), all comment becomes superfluous: see Fig. 12, see Fig. 18.

One further tip: in diagrammatic representations, the right hand mouse button conceals not only the zoom functions, but also the options for the insertion of fixed or changing data markers, together with the output of the associated curve value at the selected frequency in small additional windows. You should try out these options yourself!

Table 2:

	50Ω line	10Ω line	17Ω line	120Ω line
Electrical length in deg. at 2100Mhz		22.63	26.8	31.51
Width	1.83mm	14.53mm	7.90mm	0.25mm
Length		5.07mm	6.13mm	8.20mm

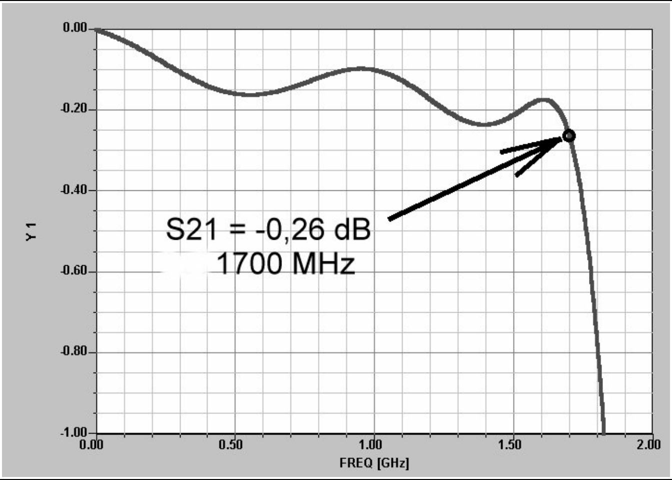

Fig 18: Results of simulation showing S21 curve.

3. Repetition of design using ANSOFT Serenade

3.1. Assembly of filter circuit

For this, go right back to the beginning and begin as if you knew absolutely nothing about the preceding design using PUFF. Just avoid duplicating work by taking all the data already determined which are valid for both designs from Sections 2.1 to 2.3.

First step:

Use a filter program to determine the coils and capacitors required for the corresponding LC low pass. Fig. 1 gives the following values for this: 2.147pF, 6.419nH, 3.698pF, 6.419nH, 2.147pF

Second step:

Use the approximation formulae provided to determine the data for the line sections for a design frequency of 1,700MHz. (Do you remember ? We didnt get involved with optimisation with PUFF until we reached a frequency of 2,100MHz, so we ignore that here too). In accordance with the selected individual impedance level, the following values were obtained from Table 3.

Third step:

Use the TRL85 calculator to determine the physical dimensions of the individual line sections, and also include the cover distance of 13 mm in the list of given printed circuit board data:

Material:	**Rogers R04003**
Board thickness:	**H = 32 MIL = 0.813 mm**
Dielectric constant:	**εr = 3.38**
Loss factorr:	**TAND = 0.001**
Copper coating:	**TH = 35 µm**
Surface roughness:	**RGH = 5 µm**
Distance between cover and PCB:	**HU = 13 mm**

The associated simulation (Synthesis) for the first line section, with 17Ω at the

Table 3:

	50Ω line	10Ω line	17Ω line	120Ω line
Electrical length at 1700MHz		22.63	22.32	32.74

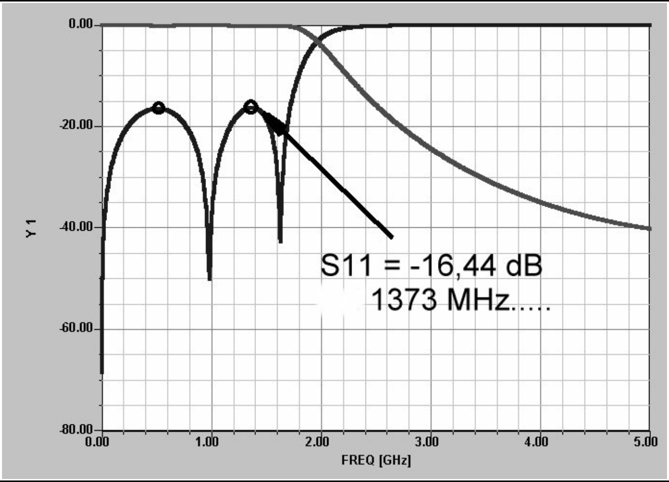

Fig 19: Results of simulation showing S11 curve.

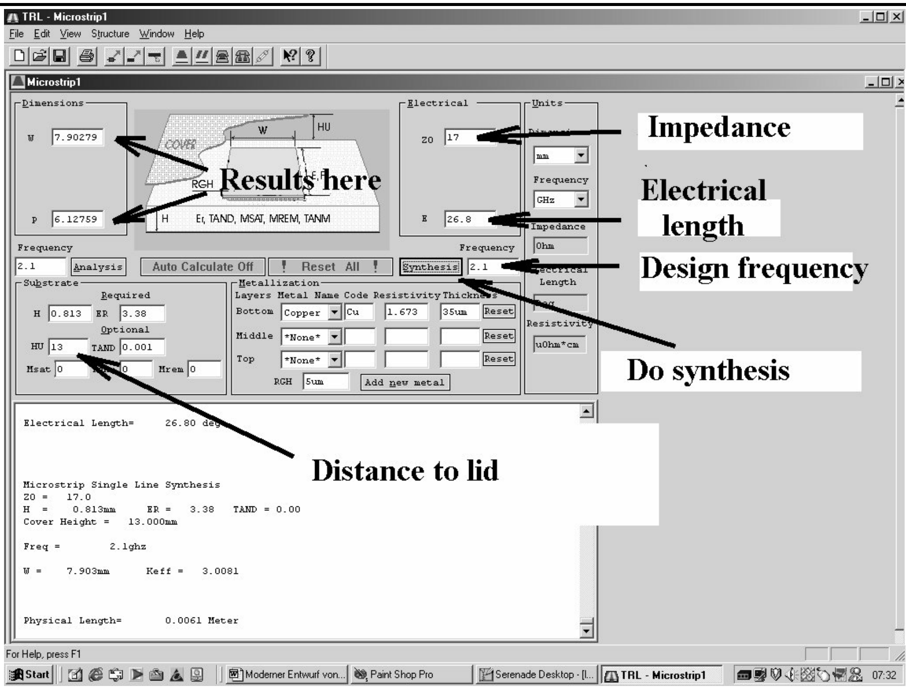

Fig 20: Setting up data for simulation taking distance to cover into account.

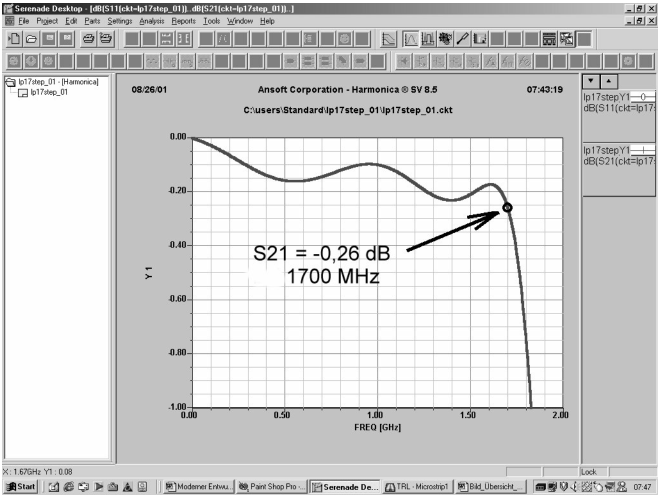

Fig 21: Results of simulation taking into account distance to cover, showing S21 curve.

frequency of 1,700MHz, shows Fig. 22. The remaining results have been entered directly into Table 4.

Fourth step:

Now we must tackle the irregularities by altering the conductor widths (Impedance Steps). Unfortunately ANSOFT was a bit mean here and blocked this option in the student version (and who just happens to have $ 20,000 to spare for the full version?). So we have to go back to the formulae in Section 2.3 and determine the substitute components using the pocket calculator. Fortunately, the conductor widths with and without the housing cover do not differ very much, so the component values can be transferred from the section referred to (quite literally, its only in the third decimal place that there is a small difference!):

Transition from	**50 Ω to 17 Ω**	**17 Ω to 120 Ω**	**120 Ω to 10 Ω**
Capacity	**0.1pF**	**0.125pF**	**0.227pF**
Inductance	**0.327nH**	**0.565 nH**	**0,655 nH**

Fifth step:

A new project is set up and then the circuit is assembled, using the physical line model. But for the length specifications of the lines we enter not values, but variables, to make optimisation easier. The following classification applies here:

- 17Ω line: Line1
- 120Ω line: „Line2
- 10Ω line: Line3

Here we access the variables block

Table 4:

	50Ω line	10Ω line	17Ω line	120Ω line
Electrical length in deg at 1700MHz		22.63	26.8	31.51
Width	1.83mm	14.53mm	7.89mm	0.25mm
Length		6.27mm	6.31mm	10.52mm

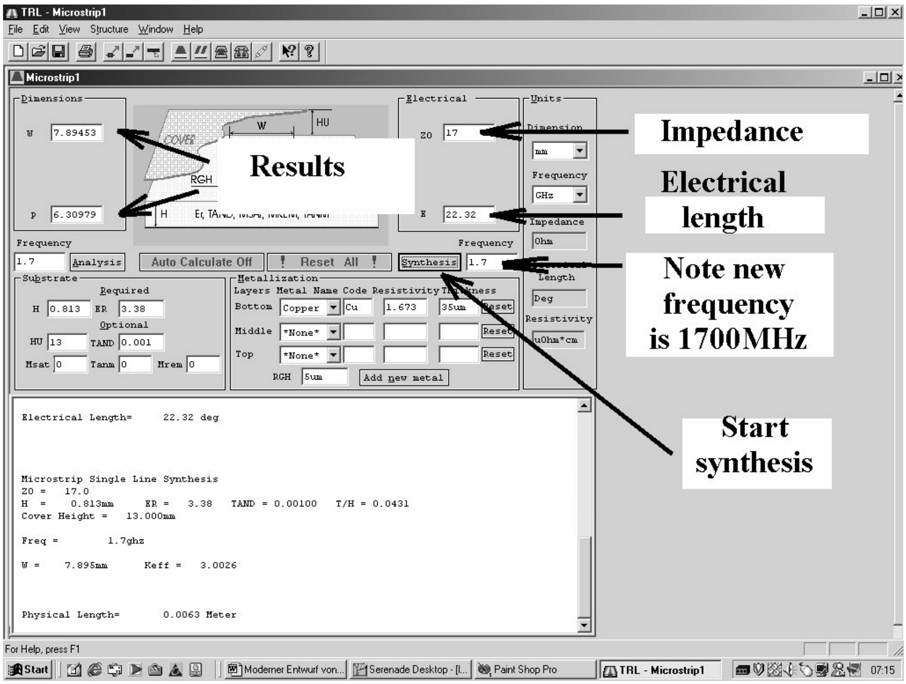

Fig 22: Setting up simulation for repitition of PUFF design.

through the "Parts" button, followed by "Control Blocks" and finally "Variables". Double click on this block to show the entries required. The lowest limiting value permitted, the target value and the upper limiting value for optimisation must always be entered between question marks (Fig. 23). In Fig. 24 we can finally see the complete circuit, ready for simulation, together with a substrate control block, a frequency control block, a variables block and an optimisation block.

Sixth step:

Now the simulation is carried out and then the results are analysed (Fig. 25 and Fig. 26). The result is exactly the same as the simulation with PUFF from Figs.11 and 12. Because of the components causing irregularities, the ripple cut off frequency has fallen to 1,300MHz and the two camels humps in S11 are of different heights.

3.2. Adjusting components for optimisation of design

Here we basically proceed as for the design with PUFF. In the first run, we vary the length of the first and last line sections, together with that of the narrow lines acting as inductances. The tuning mode is available for such purposes. The following conditions apply:

- Extending Line1 lifts the right hump, S11, in particular.
- Shortening Line2 lifts the left hump, S11, in particular.

In this manner, we can bring the maximum values of the two S11 camels humps to a value of -16.44 dB. Finally, we increase the cut off frequency by

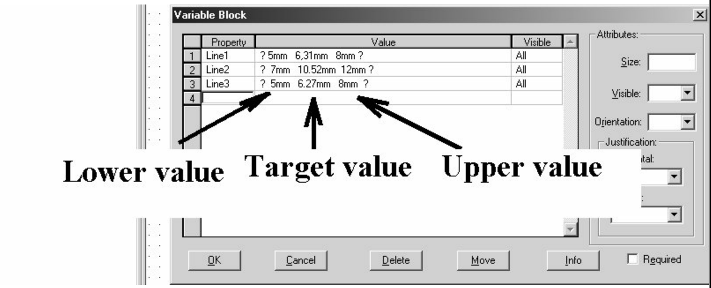

Fig 23: Setting up variables for simulation.

means of simultaneous reduction of all line lengths by the same factor to the required 1,700MHz.

Now we begin, for example, with the two outer 17Ω line sections and their length Line1. First we call up the circuit diagram onto the screen and execute the following checklist exactly (Fig. 27):

- Check that the Accumulate button has been activated.
- Click on the Tune button, which calls up the Tune menu.
- Double click on Variables Control Block in the circuit diagram to ensure that the three variables (Line1 to Line3) are transferred into the tuning list. Then click on Line1 to select it.
- Switch over to Sweep
- Delete the tick for Step Width in %
- The start value (6.5mm selected) , the final value (8mm selected) and the step width of 0.5mm can be entered in the corresponding fields.
- We are ready, we can press Tune. But first, you should call up the window showing the representation of S11 and S22 to the foreground,

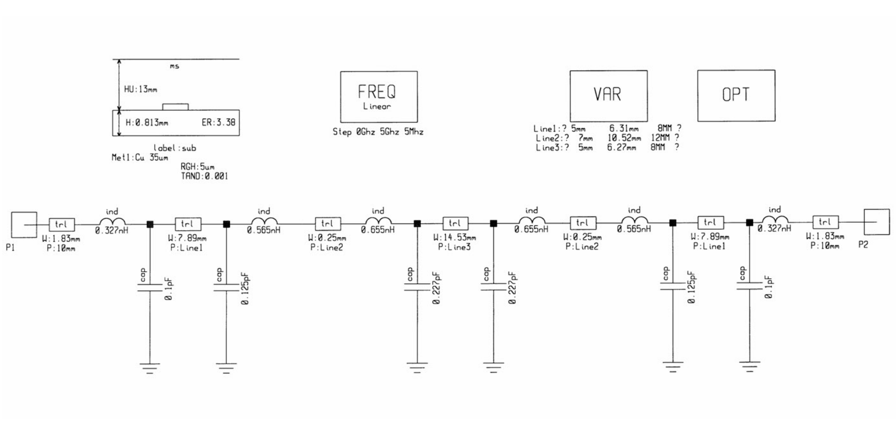

Fig 24: Completed circuit ready for simulation.

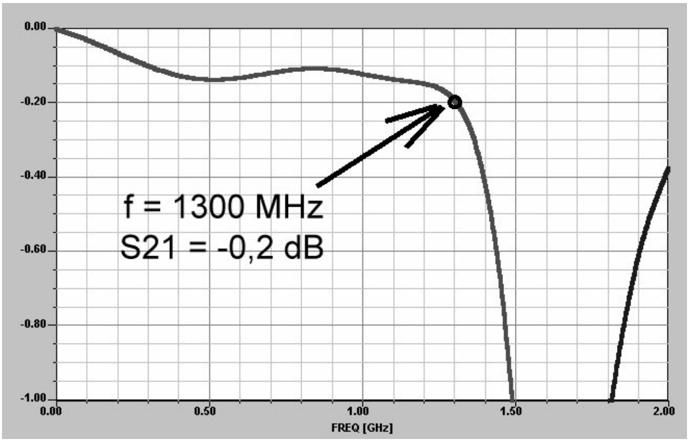

Fig 25: Results of simulation showing S21 curve.

this is very useful for looking at individual simulations and classifying them.

We can see immediately from Fig. 28 that a value of Line1 = 7.5 corresponds almost exactly to the required specifications. It is thus entered into the list as a new target value by means of double clicking on Variables Control Block and then the simulation is repeated. This time we are dealing with the precise value of the present limiting ripple value, which can be taken from Fig. 29 as S21 = -0.26 dB at 1,380 MHz. This is 320MHz too low and so we use our pocket calculator in order to shorten all line lengths by the same factor. The relationship which applies is:

$$I_{NEW} = \frac{I_{OLD} \cdot 1700}{(1700 + 320)}$$

and thus we obtain the following values, which are immediately entered into the variables control block again:

- Line1 = 6.31 mm
- Line2 = 8.85 mm
- Line3 = 5.28 mm

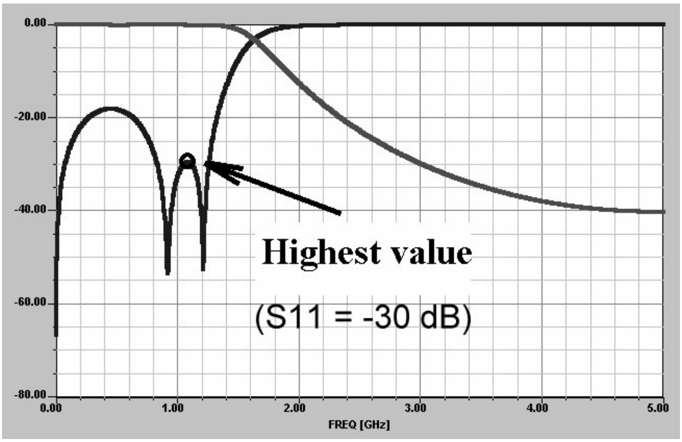

Fig 26: Results of simulation showing S11 curve.

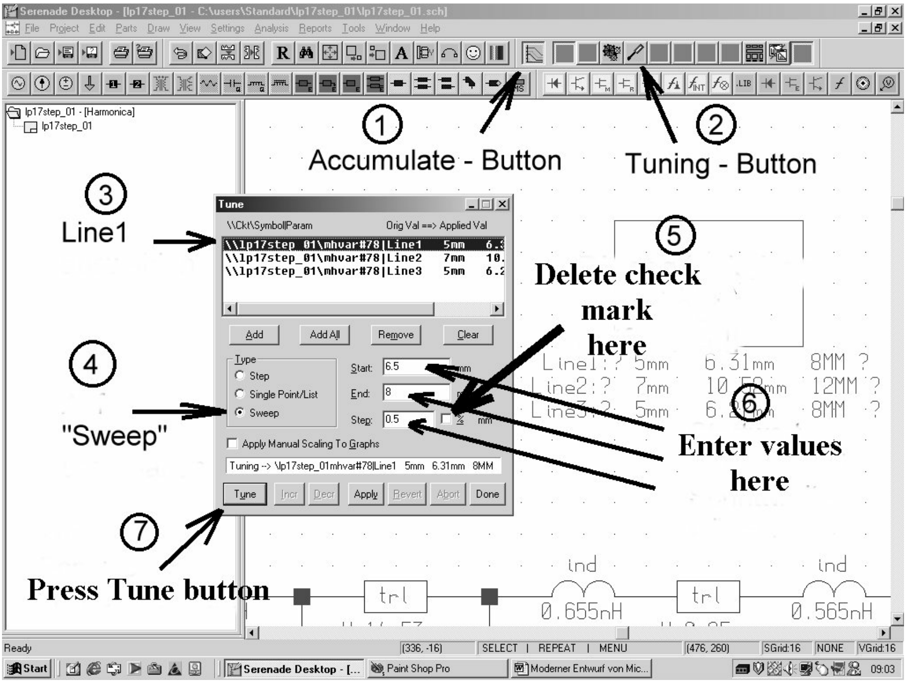

Fig 27: Setting up for tuning process.

A new simulation immediately provides information on the effect of these changes. It can be seen that here we have a cut off frequency of 1,620MHz and that a further correction of 80MHz is required. (One striking factor here is that the design frequency of the lines is now obtained at 2,100MHz, which corresponds exactly to the design using PUFF).

So now we have the following line

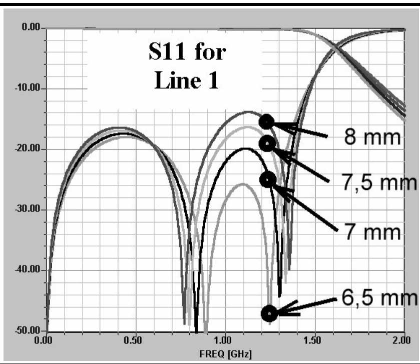

Fig 28: Results of tuning showing S11 curve.

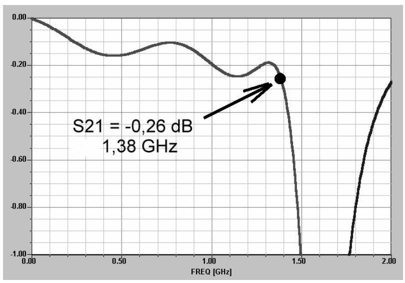

Fig 29: Results of tuning showing S21 curve.

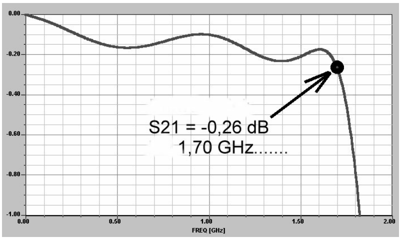

Fig 30: Final simulation results showing S21 curve.

lengths for the next stage:

- Line1 = 6.07 mm
- Line2 = 8.52 mm
- Line3 = 5.08 mm

If we use these for a simulation, we already have a cut off frequency of 1,670MHz. Yet we should look at the shape of S11 again, for these corrections have changed the hump amplitudes again, by 0.5dB. So Its clear what must be done next:

First correct the shape of S11 through Line1 and (if necessary) Line2, until you arrive at the desired hump amplitude of approximately -16.5dB. Then check the magnified shape of S21 to find the current value of the limiting ripple frequency (rated value : -0.26dB at 1,700MHz) and change all the length lines again, if necessary, by the same correction factor. If you are not satisfied, repeat this procedure again. Here, as a check and as an example, is the optimisation result for the printed circuit board design (Table 5).

Fig. 30 shows the associated final simulation of S21 as a proof of the successful design. In Fig. 31, on the other hand, we can see a sweep from 0 to 10GHz. It shows the required shape for S11 and demonstrates what should never be forgotten in relation to these microstrip line filters, the attenuation rises to a maximum and then falls again (the technical expression is scatter resonance). In principle, this is unavoidable when line sections are used in filters, and can, for example, be improved by inserting a further low pass filters with a higher cut off frequency in series.

Finally, it is very interesting to compare this design with the one using PUFF from the first part of the article. It can actually be seen that the differences in the line lengths for all designs amount to only a few hundredths of a millimetre, and it is therefore certainly the time to prepare some test boards. Their measured data (with and without housing) will then decide on any further steps necessary to make the design ready for production. Normally, an accuracy of approximately 2% can be obtained in the first stage by applying the design principles of this article.

3.3. The Optimisation Option using HARMONICA

There is also the option of carrying out this optimisation entirely by a program. To this end, for specific frequencies or frequency ranges, a minimum value or a maximum value can be entered for several parameters.

It would also be entirely conceivable to forecast as many points as possible on the S11 curve and / or the S21 curve for this filter. Then carry out about 1,000 tests, and thus to get the computer to sort out the problem, which in certain circum-

Table 5:

	50Ω line	10Ω line	17Ω line	120Ω line
Width	1.83mm	14.53mm	7.89mm	0.25mm
Length		5.08mm	6.14mm	8.20mm

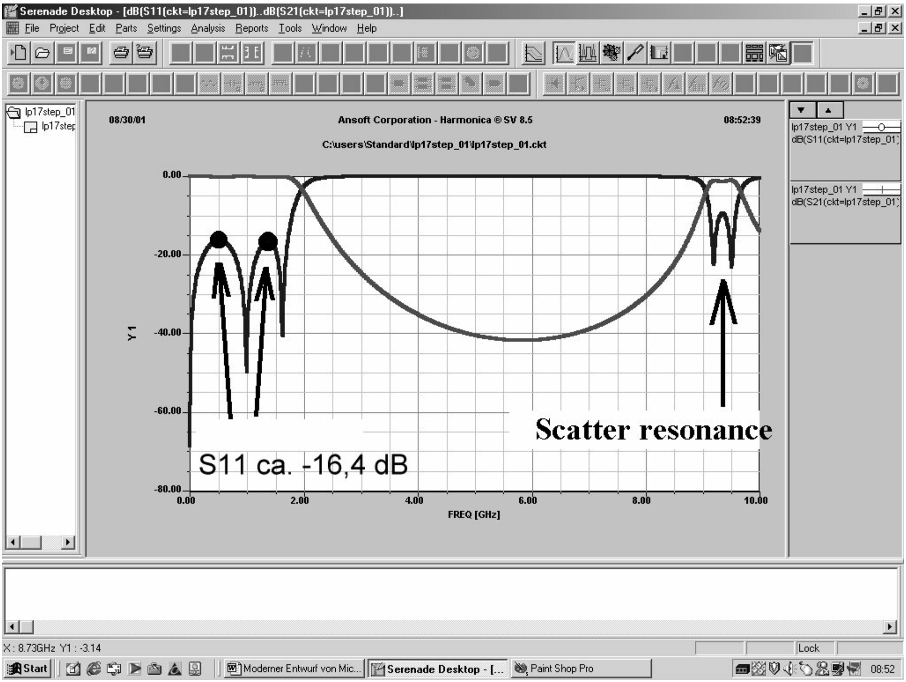

Fig 31: Sweep from 0 to 10GHz showing scatter resonance.

stances can be done within hours.

Unfortunately a maximum of 3 optimisation object are possible using the student version, and that just isnt enough. A program with so few basic points to work on quite often bends the filter curves in unpredictable or impermissible ways during the optimisation. So there would have to be a lot more objects for this to work successfully. If anyone would like to know more about this all the same, this is how to start on your way:

First step:

This would be to enter the minimum value, the target value and the maximum value for the three line lengths in the variables block. They must be entered between question marks. Luckily, that has already been done some time ago.

Second step:

Double click on the Linear Optimisation button (=black yellow red target) and open the sheet with the target objectives (Fig. 32) and enter the following conditions correctly:

Objective 1:

Make sure that in the frequency range between 1MHz and 1,700MHz the amplitude of S21 does not fall below -0.26dB.

Objective 2:

Make sure that in the frequency range between 1MHz and 1,500MHz the amplitude of S11 always remains below -16.5 dB.

Objective 3:

Make sure that at 1,700MHz precisely an amplitude of -0.26dB can be measured at S21.

Tip: Please avoid the value Zero MHz

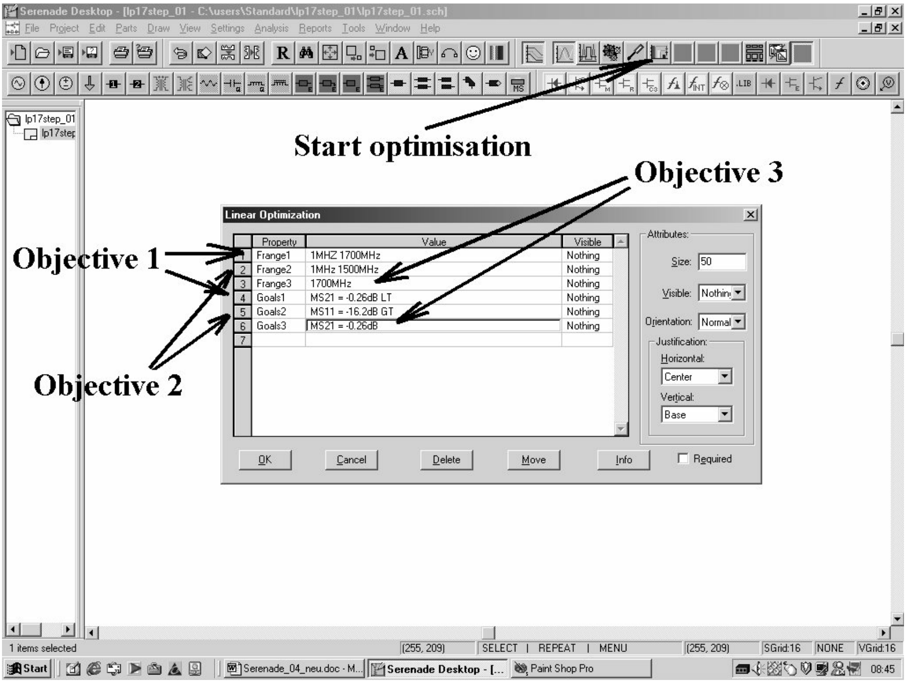

Fig 32: Setting up objectives for optimisation.

when entering frequencies, since this will lead to an error message!

Then close the Objective menu and press the button to start the optimisation.

Third step:

Please follow the sequence listed in Fig. 33 exactly and to do this call up the (blue) diagram with the simulation results for S11 and S21 into the foreground for observation. Then close the window again with Close. Hopefully everything has gone all right as described, and a usable result has been obtained.

4. Measurement results

Both design methods produce a prototype built into a milled aluminium housing, fitted with SMA flanged sockets, and finally measured with a modern network analyser. The results are presented in the original untouched condition, for we are actually trying to make sure that the simulation can produce something that agrees with reality.

Fig. 34 shows the measured transmission range for the design using PUFF (amplitude range between 0 and -1 dB). Several things immediately strike one by comparison with the simulation:

The first Chebyschev ripple has a higher amplitude than the second

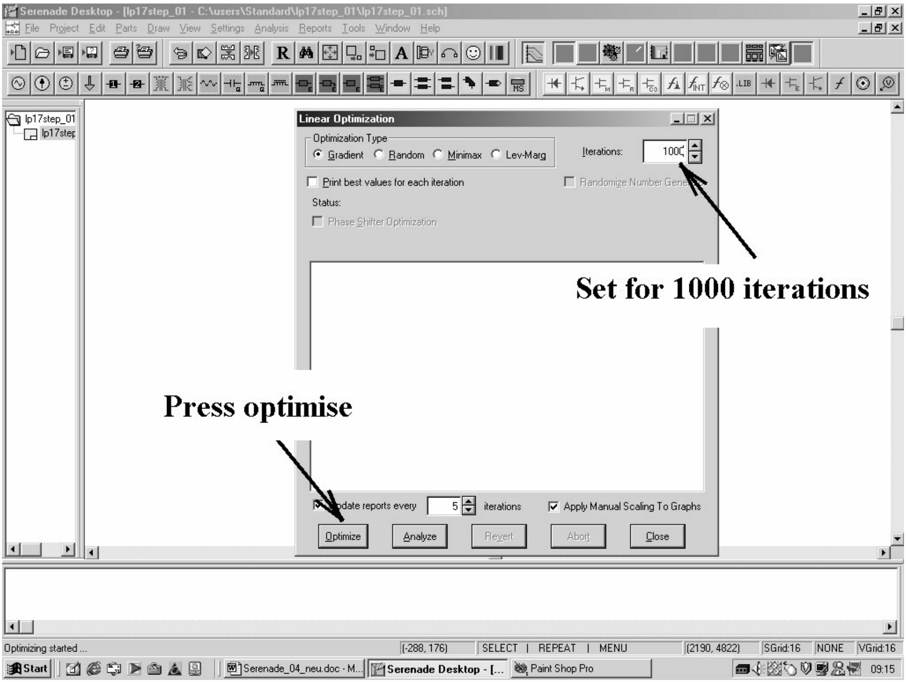

Fig 33: Final set up to satrt optimisation.

The cut off frequency, at 1.76GHz, is a good 3 % higher than forecast

The fundamental filter attenuation is markedly lower than was expected

Now lets look at these points in order:

In Fig. 35 we can immediately see the reasons for the unequal ripples observed in a). The two camels humps of S11 are not the same height! (If we look at the literature again, we see where the root cause lies> The formulae for the replacement components for the impedance steps are no longer accurate enough when there are extreme differences in the widths, such as 0.25mm to 14.6mm. Observant readers will realise at once that the lengths of the two external capacitor lines must be increased somewhat for correction purposes. This automatically lowers the cut off frequency, so that both objections, a) and b), have disappeared afterwards.

The low fundamental filter attenuation is also interesting. It has its origins in the fact that the loss factor decreases markedly as the frequency falls, which is why a comprehensive data sheet was requested from the German importer. This contains an interesting diagram, showing the insertion attenuation of a microstrip line for various frequencies, in which various ROGERS materials are used one after another (Fig. 36). If we analyse these curves very precisely, for the R04003 material, then we can come to the following conclusions:

The loss factor tanδ, that is 0.002 at 10GHz, will fall by a factor of approximately 5.3 if the operating frequency is reduced to 1.5 to 2GHz.

If we now repeat the simulation with the new value of tanδ = 0.002 / 5.3 =

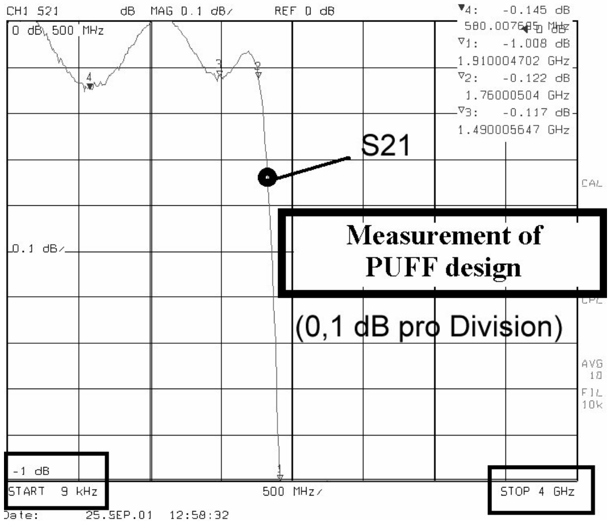

Fig 34: Measurement results on PUFF design.

0.00038, then we obtain, very accurately, the attenuation observed during the measurements. (From now on, this must naturally be taken into account in future designs).

There is thus almost nothing more to say with regard to Fig. 37 and Fig. 38, for the ANSOFT design which supplies only a slightly lower ripple in the transmission range and smaller discrepancies for the camels humps of S11. Both the essential shape and the fundamental attenuation and the cut off frequency show scarcely any difference from the design using PUFF, so that what has just been said is also true for any further procedures using the ANSOFT printed circuit board.

The old adage from developers experi-

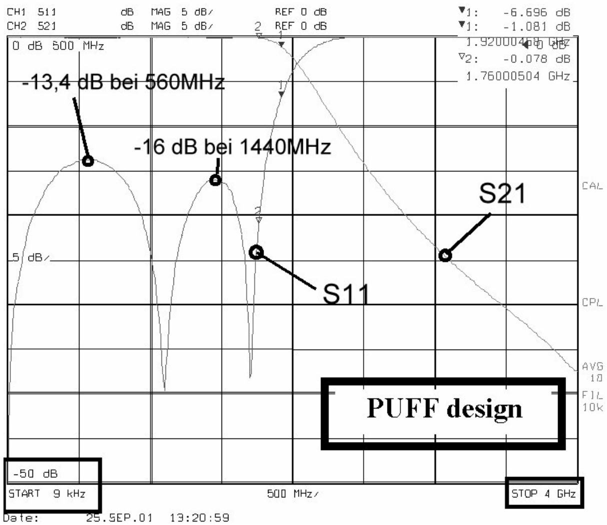

Fig 35: Measurement results on PUFF design showing unequal humps.

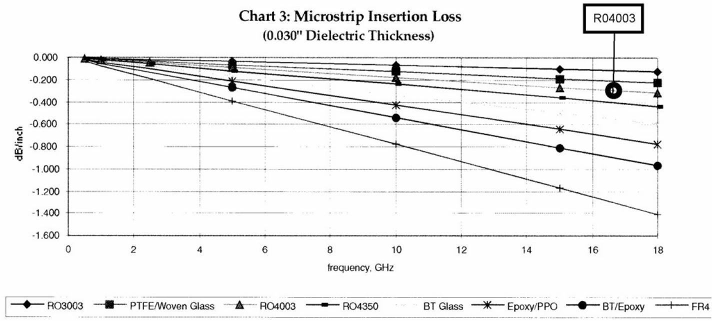

Fig 36: Data sheet on Rogers material.

ence is also confirmed in this case, that its not until the third board design that youre ready for full production.

5. Summary and Evaluation

PUFF and ANSOFT HARMONICA are equally suitable for filter design. Those who prefer to trust their instincts and their experience as regards the optimisation which is required, or who want to test out spontaneous ideas as fast as possible, or who work with such programs only occasionally, are usually going to achieve their objectives faster using PUFF.

However, as soon as housing influences are brought in, or if many parameters are varied, or if various zoom functions are used, if markers are superimposed or if all WINDOWS options are to be used,

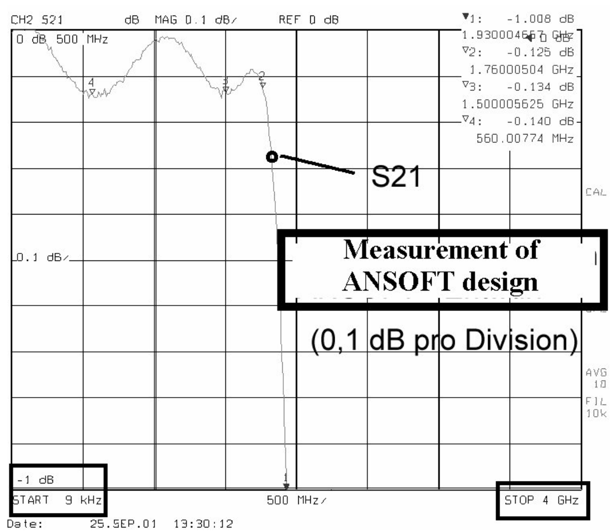

Fig 37: Measurement results on ANSOFT design.

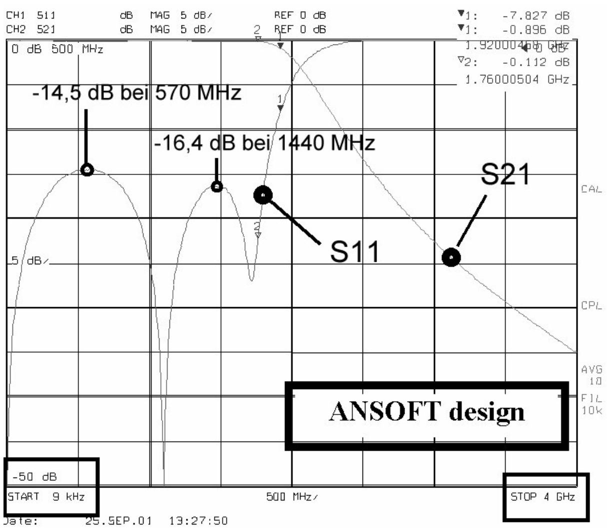

Fig 38: Measurement results on PUFF design.

you are naturally much better off with the student version of Serenade. It is simply a very modern development tool, accurately tailored to the practical needs of radio frequency developers. If only those all important options currently blocked were also accessible, then there would be no question at all about what to use.

6. Literature

[1] Gunthard Kraus: Modern Design of Stripline Band-passes VHF Reports, issue 2/2001

[2] English. Original Manual for PUFF

[3] Manual for ANSOFT-Serenade, (included in program for download)

[4] K. C. Gupta, Ramesh Garg, Inder Bahl, Prakash Bhartia: Microstrip Lines and Slotlines. Artech House Publishers Boston and London 1996. ISBN-Nr.: 0-89006-766-X

[5] David Pozar: Microwave Engineering. John Wiley and Sons, New York 1998. ISBN Nr.: 0-471-17096-8